Available online at www.sciencedirect.com

ScienceDirect

Procedia Engineering 126 (2015) 701 – 705

Procedia Engineering

www.elsevier.com/locate/procedia

7th International Conference on Fluid Mechanics, ICFM7

Numerical investigation of film cooling using RANS and LES

Shuo Li, Jiemin Zhan, Yejun Gong*, Wenqing Hu

Departmentof Applied Mechanics and Engineering, Schools of engineering, Sun Yat-sen University, Guangzhou 510275, China

Abstract

Film cooling of turbine blades is an effective way to cool the blade and ensure the long life of turbine blades. Numerical simulation of the film cooling process requires the accurate prediction of the vortex flow in the near wall region. This study compared several popular turbulence models, the realizable k-ε model, the k-ω SST model and the LES model. The dimensionless temperature distributions are compared for these turbulence models, and only LES is able to capture the small detachment of the main flow from the turbine blade surface. More details of the pressure, temperature, vorticity and film cooling effectiveness profile are shown in streamwise and spanwise direction for the LES case. Film cooling becomes weaken once the flow detaches from the blade surface.

Peer-review under responsibility of The Chinese Society of Theoretical and Applied Mechanics (CSTAM)

*Keywords:*gas turbine, film cooling, turbulence model, LES

1. Introduction

Developments in turbine cooling technology play a crucial role in the improvement of the thermal efficiency and power output of advanced gas turbines. Film cooling of turbine blades is an effective way to cool the blade and ensure the long life of turbine blades. For the high temperature gas turbine, their hot blades and vanes are cooled by injecting low temperature air through the internal coolant passages on the blade surface in order to form a protective layer between the blade surface and the hot gas path. The interaction between the film cooling air and the main flow forms a shear layer that leads to the mixing and decay of film cooling along the blade surface. The cooling air penetrates into the main flow through a vertical hole and generates vortex structures via the interaction between the main flow and the film cooling jet. Then the cooling air is heated, and the efficiency of film cooling on the turbine blade surface is decreased in the downstream of the main flow.

Numerous experimental studies [1-3] and numerical investigations [4-6] have been conducted on film cooling. A review paper by Goldstein [7] summarized the early works in this area. Bogard and Thole (2006) reviewed the

Corresponding author: Tel:13535543013
Email address:gongyj3@mail.sysu.edu.cn

Peer-review under responsibility of The Chinese Society of Theoretical and Applied Mechanics (CSTAM)
doi:10.1016/j.proeng.2015.11.274

recent progresses [8]. Most of the previous numerical studies are built on the Reynolds-averaged Navier-Stokes (RANS) method, where the most used turbulence model is the two-equation turbulence model (k-ε or k-ω). A systematic study of film cooling by Demuren et al. [9] revealed that the complex flow field behind the jet was not properly resolved and the turbulent mixing process was simulated crudely using the eddy viscosity model. Demuren [10] carried out computations using a multi-grid method and used a second-moment closure model to approximate the Reynolds stresses. Although a fairly good prediction of mean flow trends was reported, there was considerable uncertainty regarding the accuracy of jet penetration height. The uncertainties in the previous investigations motivate the present study. Several existing turbulence models are used to predict the film cooling effectiveness, and their performances are evaluated and compared.

2. Problem description

The problem configuration is based on the experimental setup in the study of Liang Jun Yu [11], where cool air is injected through round holes with 90 degree of jet angle. The whole film cooling process is described in Fig.1. One single jet is positioned on the bottom of the rectangular-parallelepiped-shaped shell, i.e. the film cooling hole in Fig. 1. The diameter of the hole is D=12mm. The guide channel for the exit of the film-cooling hole was perpendicular (inclined at 90 degrees) toward the main flow direction. The rectangular-parallelepiped-shaped shell represents the wind tunnel with an open circuit. The coolant flow was created in a cavity located in the plenum before entering the film-cooling hole. The inlet of high temperature main flow gas located 19D upstream to the leading edge of the hole, while the outflow plane located 30D downstream to the trailing edge of the hole. The blowing ratio $M=\rho_j U_j/(\rho_\infty U_\infty)$ is set up as 0.045, where U_j is the jet velocity and U_∞ is the initial main flow velocity. The initial temperature of the coolant is 188 K, while the main flow is 285 K. The blade surface is set up as an adiabatic wall. Figures 2 and 3 show the computational domain with approximately 10^6 cells using multi-block parametric grid generation. The flow characteristics in this study will be predicted by several different turbulence models.

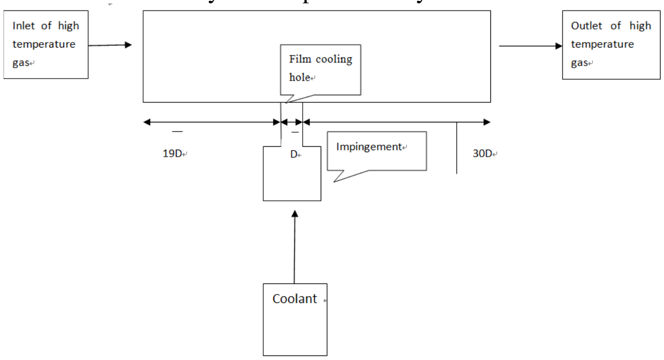

Fig.1 Problem configuration

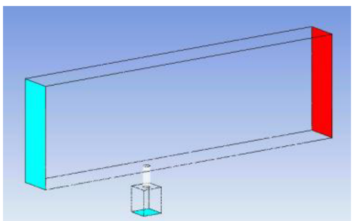

Fig .2 Mesh structure

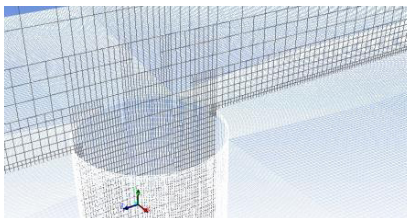

Fig.3 Near wall mesh structure

3. Mathematical modeling

In the present study, the working fluid is assumed to be incompressible and Newtonian with temperature-dependent fluid properties. The accuracy of the prediction is based on the ability of the closure expressions in capturing the flow physics. Both RANS (the realizable k-ε model, k-ω SST model) and large eddy simulation (LES) turbulence models are used in this study. Turbulence effects are taken into account using the eddy viscosity/diffusivity concept. In the LES approach, the large, energy carrying, dynamically important, and flow-dependent eddies are solved directly, leaving only small scale of turbulence with very low energy and supposedly universal behaviour to be modelled. To accurate simulate the flow in the near wall region, particular near wall treatment was utilized in this study. Turbulent flows are substantially affected by the presence of walls. A high quality mesh for LES method should have enough grids inside the boundary layer, which means a huge mesh size demanded. Based on the work of Werner and Wengle [12], an analytical integration of the power-law near wall velocity distribution is employed. With this treatment, the calculation accuracy is maintained while the mesh is simplified. The near wall mesh structure was shown in Fig.3, where adaptive mesh refinement is used to meet the demand of small enough wall cell.

4. Results

The governing equations were resolved using a commercial CFD code, ANSYS FLUENT with second-order central difference scheme for the viscous terms and the power law based scheme for the convective terms.

Fig. 3 shows the contour plot of dimensionless temperature, which is defined as

$$T_{\mathrm{dim}} = \frac{T - T_\infty}{T_j - T_\infty}, \tag{1}$$

where T is the local static temperature, T_j and T_∞ are the temperature of the jet flow and the main flow, respectively. On the adiabatic wall, T equals the wall temperature, T_{aw}, and we can rewrite Eq. 1 as below,

$$\eta = \frac{T_{aw} - T_\infty}{T_j - T_\infty}, \tag{2}$$

where η denotes the adiabatic effectiveness. The contour plot of T_{dim} in Fig. 4 reveals the differences in three turbulent models, the realizable k-ε model, k-ω SST model and LES model. The dimensionless temperature T_{dim} isthe converged result after 100,000 time-steps with steps size of 0.0005s. The k-ε method predicts a clear thermal gradient lift-off from the surface, the k-ω SST shows more tortuous thermal gradient at the jet spreading direction, and the LES predicts more mixing between the jet and main flow in vertical directions, which is captured in the experimental study by Liang [11]. Obviously, only LES captures the highly unsteady features resulting from the interaction between the jet flow and the main flow, and the detachment of the jet flow from the blade surface, which is also observed in experiments by Liang [11]. This detachment may lead to the low effectiveness locally and temporarily. One possible reason is the high turbulent intensity near the wall.

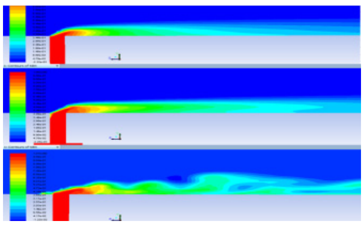

Fig.4.Colour plot of dimensionless temperature at the y-z plane .Top to bottom: realizable k-ε, k-ω SST and LES

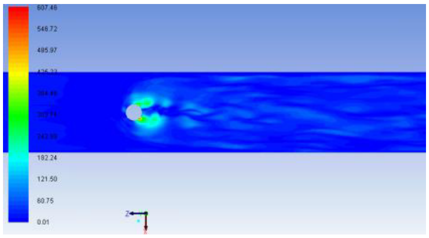

Fig.5. Dynamic pressure distribution on the turbine blade using LES

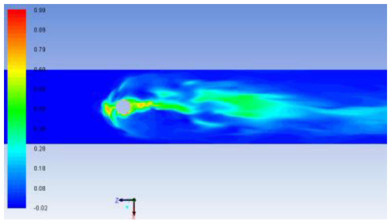

Fig.6. Dimensionless Temperature distribution on the turbine blade using LES

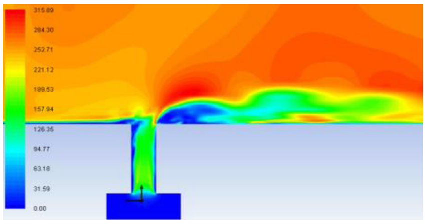

Fig.7. Dynamic Pressure distribution at the y-z plane using LES

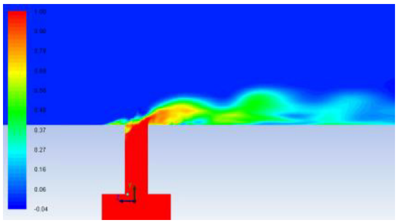

Fig.8. Dimensionless Temperature distribution at the *y*-*z* plane using LES

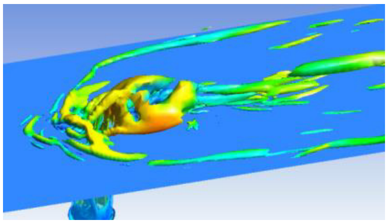

Fig.9. Iso-surface of vorticities using LES

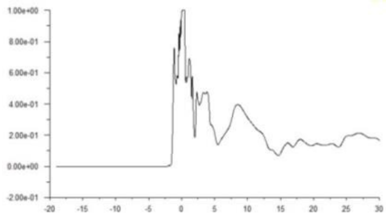

Fig.10. Dimensionless temperature vs dimensionless *z* using LES

Figures 5 and 6 show the dynamic pressure and dimensionless temperature distribution on the turbine blade. In Fig. 5, a pair of high dynamic pressure areas was observed near the hole, which indicates high velocity near the hole. Fig.6 shows jet temperature drops quickly in spanwise direction. This illustrates that the heat transfer is not only unsteady in spanwise direction, but also quickly dilutes in streamwise direction. Furthermore, the temperature difference is also high near the hole, i.e. film cooling is more effective near the hole. Figures 7 and 8 show the spanwise dynamic pressure and temperature, and it is clear that how the flow motion affects the film cooling process near the wall. In Fig.8, the film cooling is most effective in a small area near the hole which is in accordant with what observed in Fig.6. More details of the instantaneously vortex structures simulated by the LES model are displayed in Fig. 9. The vortex structure in the spanwise direction is also captured, though quickly dispersed in the streamwise direction. Furthermore, Fig.10 plots the dimensionless temperature on the blade surface along the spanwise direction, and it is obvious that the film cooling is highly effective near the hole (z=0), and becomes weaken when z>15 (z is the dimensionless distance from the hole centre in spanwise direction).

5. Conclusion

In this study, the evolution of the cooling injected jet flow has been discussed using different turbulent models. The temperature contour plot obtained from LES shows better estimation of the film cooling. Compared with the RANS models, LES method is able to resolve the detailed vortex structures, and performs better in the prediction of vortex flow in the near-wall region where the injected jet interacts with the main flow. Furthermore, only LES is able to capture the detachment of the main flow from the turbine wall, which leads to the weakening of the film cooling effectiveness.

In conclusion, LES is necessary for the accurate prediction of film cooling effectiveness, due to the high gradient temperature and complex vortex structure near the turbine blade surface.

References

[1] Fric, T.F.,and Roshko, A., Vortical structures in the wake of a transverse Jet, J. Fluid Mech., 279(1994)1–47.

[2] Lee, S.W., Lee, J.S., and Ro, S.T., Experimental study on the flow characteristics of streamwise inclined jets in crossflow on flat plane, ASME J.Turbomach.,116(1994) 97-105.

[3] Kelso, R.M.,Lim, T.T., and Perry, A.E., An experimental study of round jets in crossflow, J. Fluid Mech, 306(1996) 111–144

[4] Garg, V. K., and Gaugler, R. E., , "Effect of coolant temperature and mass flow on film cooling of turbine blade, Int. J. Heat Mass Transf., 40(1997) 435-445.

[5] Kim, S. W., and Benson, T.J., Fluid flow of a row of jets in crossflow-A numerical study, AIAA J., 31(1993) 806-811

[6] Walters, D.K., Leylek, J.H., A detailed analysis of film-cooling physics: Part I-streamwise Injection With Cylindrical Holes, ASME J. Turbomach., 122(2000) 102-112.

[7] Goldstein, R.J., Film cooling, Adv. Heat Transf., 7(1971) 321-379

[8] Bogard, D.G., Thole, K.A., Gas turbine film cooling, J. Propul. Power, 22 (2006) 249-270

[9] Demuren, D.K., Rodi, W., Schonung, B., Systematic study of film cooling with a three-dimensional calculation procedure, ASME J. Turbomach., 108(1986) 124-130.

[10] Demuren, A.O., Characteristics of three-dimensional turbulent jets in crossflow, Int. J. Eng. Sci., 31(1993) 899-913

[11] Liang Jun-Yu, Meng Bao-Bao, Kang Shun, Zhai Li-Na, Experimental study of the turbulent flow downstream of film cooling holes on a flat plate, Journal of Chinese Society of Power Engineering, , 32(2012):435-439.

[12] H. Werner, H. Wengle. Large-eddy simulation of turbulent flow over and around a cube in a plate channel. In Eighth Symposium on Turbulent Shear Flows, Munich, Germany. 1991.